FLOW FIELD ANALYSIS OF TWO DIMENSIONAL SUPERSONIC INTAKE

A Thesis Submitted
In Partial Fulfilment of the Requirements
for the Degree of
MASTER OF TECHNOLOGY

by

VINAYAK SISTA

to the

DEPARTMENT OF AEROSPACE ENGINEERING

INDIAN INSTITUTE OF TECHNOLOGY, KANPUR

MAY, 1989

CERTIFICATE

This is to certify that the thesis work entitled, " Flow Field Analysis of a Two Dimensional Supersonic Intake" has been carried out by Sista.Venkata.Vinayaka.Vishwanatham under my supervision and that it has not been submitted elsewhere for a degree.

O P Sharma

Dr. O.P. SHARMA
Professor
Department of Aeronautical Engg.
Indian Institute of Technology
Kanpur.

CONTENTS

NOMENCLATURE

τ = Ratio of specific heats = 1.4

u = Velocity component in x direction

v = Velocity component in y direction

p = Pressure

T = Temperature

R = Charecteristic gas constant

U,F,G = Fluxes in conservation form

h_o = Stagnation or total enthalpy

e = internal energy

A = Area of the C

H_{AB}, H_{BC}, H_{CD}, H_{DA} = Are the fluxes through the sides

x_A, x_B = x coordinates at A and B

y_A, y_B = y coordinates at A and B

w = Relaxation factor

Θ = Wedge angle

β = Shock angle

M = Mach numbers

x = Step length in x direction

y = Step length in y direction

t = Time step

LIST OF FIG.

Chapter 1

GENERAL INTRODUCTION

With the advent of supersonic aircraft,the design of the intake has become critical as it has a marked effect on the engine efficiency and external drag. In the early stages, the performance was primarily predicted experimentally, that is, from wind tunnel testing. But now the availability of high speed, high storage capacity computers, and algorithmic sophistication has allowed the application of advanced analytical methods to several practical design problems including the inlet design and performance analysis.

Modern computational methods allow direct numerical solution of inlet flow field in an efficient manner. A variety of integrated performance parameters can be obtained from such a solution. Thus different configurations can be developed with less reliance on wind tunnel testing, especially in the preliminary design stages. However, computational fluid dynamics is still a growing field and for too complex flows, exact solution is not possible even with present algorithms and computing machines. In such cases, exact solutions are obtained for simpler configurations to establish correctness of the procedure and then modified to handle complicated cases.

The primary function of an inlet is to supply

continuous air flow to the engine at a desired rate with highest possible recovery of total pressure, the lowest possible external drag and minimum weight. Further it should also provide uniform flow distribution at the entry of the compressor or combustion chamber. Usually the entry Mach number is around 0.4 for subsonic combustion. For a uniform flow field and design Mach number, an intake will have maximum possible pressure recovery, but for a wide range of mach numbers and angles of attack, the performance is far from optimum. During the deceleration of airflow, the static pressure is increased. Since the process of deceleration can be achieved both inside as well as outside the inlet, it is convenient to discuss each separately.

INTERNAL COMPRESSION TYPE

Here the compression is achieved through a series of internal oblique shocks followed by normal shock positioned downstream of the throat of the intake. Finally the subsonic flow downstream of the constant area throat is further slowed down by moving through a divergent section.

EXTERNAL COMPRESSION TYPE

The compression in this type of inlet is achieved by either one or a series of oblique shocks followed by a normal shock. The external compression inlet that achieves compression through single normal shock is called Pitot inlet or Normal shock inlet. This type of inlet is simple, short and light in

weight. Total pressure recovery of this inlet corresponds to total pressure ratio across the normal shock. However for numbers greater than 1.6, The total pressure recovery is too low and a more efficient design should be used.

The external compression inlet with one or more oblique shocks has its inlet throat at or very near the cowl lip. Increasing the number of oblique shocks increases the total pressure recovery for any free stream Mach number. The compression is achieved by a ramp. This type of inlet can be used upto Mach number 2.5 with acceptable pressure ratio and cowl drag.

MIXED COMPRESSION TYPE

This type of inlet achieves compression through the external oblique shocks, internal reflected oblique shocks, and the terminal normal shock. Ideal location of normal shock is just downstream of inlet throat to minimize total pressure losses and to maintain a stable operating location of this shock. It can be used above Mach 2.5. It is more complex, heavier and costlier than external compression types.

The supersonic inlets can also be classified as two dimensional (rectangular)and axisymmetric (circular). Axisymmetric inlets have a slight advantage over two dimensional inlets with respect to weight and total pressure ratio. However, the two dimensional inlets have an advantage in design simplicity and in providing larger variation in inlet

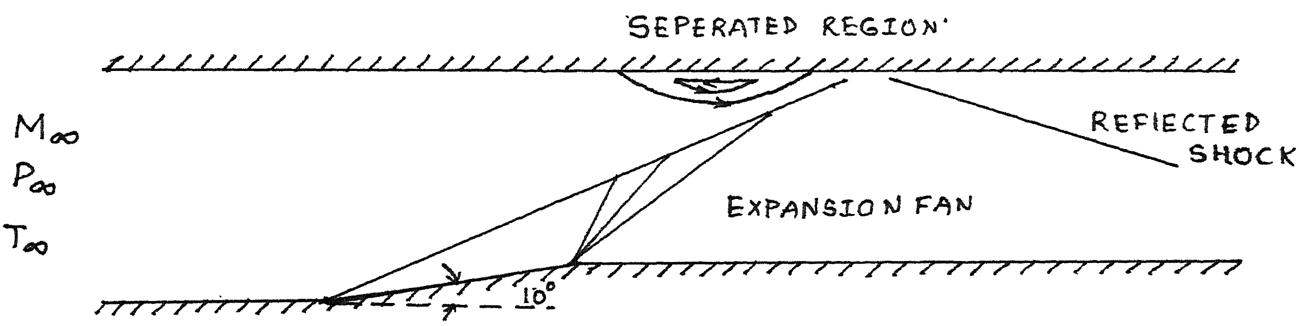

Fig: (1.1)

Sketch of supersonic Intake

the flow field for an arbitrary geometry. The geometry selected is as shown in fig(1.1). The upper wall is kept parallel to the free stream, whereas at the lower wall the flow undergoes a 10^{o} compression at 2 cm. and a 10^{o} expansion at 4 cm. Total length of the intake is 10 cm. and height is 2 cm. In this inlet we study the variations in pressure, density, and velocity across the shocks. From this we know the maximum possible pressure recovery of the inlet. It is a complex problem with shock expansion layer interaction and shock boundary layer interaction etc. So, for sake of simplicity, certain assumptions are made. These are described in the next chapter. For this problem the solutions are obtained for inflow condition as given, Mach number=5, P=1.013, and T=298K. After calculations the results are plotted and compared.

1.2. LITERATURE REVIEW

In the area of inviscid compressible flows there are a variety of practical problems. These include rocket nozzle flows, aircraft and missile inlet flows, etc. The methods that have evolved and are used currently for the solution of compressible flows fall roughly into two categories. The first is the unsteady method which is used to integrate the flow eqns. with respect to time for both steady and unsteady flows. This approach has been useful in overcoming difficulties associated with shock waves and transonic flows. The second approach is the marching techniques for integration of steady eqns. in

supersonic flows.

The use of unsteady methods for supersonic free streams is primarily for the computation of unsteady or steady flows with embedded subsonic region. Bohachevsky and Rubin (1966) applied unsteady finite difference methods to $M_\infty > 1$ flows. These authors used the highly damped method of Lax (1954), to solve inviscid Euler eqns. for flow past a blunt body. Next came Morrety and colleagues using schemes suggested by Lax and Wendroff(1960,1964) and, successively by the Maccormack method(1969). The solutions of Lax-Wendroff, Maccormack and Rusnow methods worked satisfactorily for flow past bodies with fairly regular shapes such as wedges, spheres and cones, but have a tendency to fail for complicated geometries.

For smooth flows one can use the Maccormack scheme (1981) to integrate the inviscid flow eqn. and obtain good results. For steady flows one can replace the unsteady energy eqn. by steady constant-total-enthalpy eqn. and still obtain good results. This was tested for compressible inviscid flows for different geometries and fairly good results were obtained.

Morrety (1978, 1979) set up a method that examined characteristic directions and selected difference formulas to approximate the flow eqns. depending on behavior of the characteristics. His procedure was to write the inviscid flow eqns. in the form which in characteristic methods is called compatibility eqns. If these are integrated in characteristic

direction they reduce to ordinary differential eqns., which are much simpler to deal with. He tested this scheme for flow past blunt bodies and aircraft configurations successfully. The method of characteristics gives a better solution than any other method but the drawback is the tedious procedure involved in computing.

IMPLICIT METHODS

The limitations on the time step induced by stability considerations associated with explicit schemes is often too restrictive in applications, and consequently implicit schemes are used more and more to overcome this limitation. Ritchermeyer and Morten (1967) proposed this scheme for the one dimensional case. This method has restricted use due to programming complexities and computing time and stability restrictions. Ajay Kumar (1981) developed a new method combining explicit and implicit methods and tested it on a supersonic intake model. Results were fairly accurate and there was saving in computer time.

Chapter 2

Formulation of the problem

The basic assumptions in solving the flow field of a supersonic inlet are as follows:

1. Intake is two dimensional.
2. Flow is steady, inviscid and adiabatic.
3. No body forces
4. Turbulence effects neglected.
5. Ratios of specific heats (γ) and entropy are assumed constant.

Usually axisymmetric intakes are used due to higher pressure recovery but due to design simplicity a two dimensional model has been selected. In reality the flow is viscous, hence there is formation of boundary layer, heat transfer effects near wall and many other undesirable effects. In supersonic flows shock formation will be present and in the presence of boundary layer, there will be a shock wave boundary layer interaction which may lead to separation. In the supersonic intakes the temperature also varies rapidly after the shock, which leads to variation in (γ) and rise in entropy. The analysis with the above mentioned things is too complicated, so the present analysis is restricted to steady inviscid flow.

FUNDAMENTAL DIFFICULTIES

Basically there are four dependent variables in any ideal gas, two dimensional flow problem: Two velocity components and two thermodynamic properties. In the case of an incompressible flow, the energy equation is not required to solve the momentum and continuity equations, removing one thermodynamic property, namely temperature. The pressure may be removed by cross differentiation, and vorticity may be introduced. The two velocity components are removed by introduction of stream function, leaving the two unknowns vorticity and stream function to be solved for by two equations. In case of compressible flow the energy equation is required to solve the others, and stream function is not defined for unsteady flow.So we must work with four coupled partial differential equations.

The inviscid equations for supersonic flow are hyperbolic. The most formidable numerical problem unique to supersonic flow is existence of shock waves. In high Reynold's number limit, shock waves are discontinuities in the flow solution. Most of the efforts in this area have been directed towards smearing (capturing) out these discontinuities.

The numerical computation is complicated, because the regular solution may break down due to nonlinearity of the

governing equations. The breakdown is characterized by shock wave formation. In the last decade two numerical techniques have emerged for the analysis of these flows. One is the shock capturing technique. It tries to remove explicit computation of discontinuities by generalizing the concept of a solution of an Euler equation to include weak solutions (i.e. discontinuities). As this scheme requires no special treatment to deal with discontinuities, it has become extremely popular in computing. However despite its present popularity it is a poor interpretation of the physical phenomenon and an uneconomical way of computing.

The other technique is shock fitting. It makes special provisions for explicitly computing discontinuities. In essence it locates the discontinuities and treats them as boundaries between regions where a regular solution is valid.

2.b MATHEMATICAL FORMULATION

The governing equations for an inviscid, two dimensional incompressible flow are as follows:

1. Continuity equation:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} = 0$$

2. Momentum eqn. in x-direction

$$\rho\frac{\partial u}{\partial t} + \rho u\frac{\partial u}{\partial x} + \rho v\frac{\partial u}{\partial y} = -\frac{\partial p}{\partial x} \qquad ..2.2$$

3. Momentum eqn. in y direction

$$\rho\frac{\partial v}{\partial t} + \rho u\frac{\partial v}{\partial x} + \rho v\frac{\partial v}{\partial y} = -\frac{\partial p}{\partial y} \qquad ..2.3$$

4. Energy eqn.

$$\frac{\partial}{\partial t}\left(e + \frac{V^2}{2}\right) + \rho u\frac{\partial}{\partial x}\left(e + \frac{V^2}{2}\right) + \rho v\frac{\partial}{\partial y}\left(e + \frac{V^2}{2}\right)$$

$$= -\frac{\partial}{\partial x}\left(Pu \right) - \frac{\partial}{\partial y}\left(Pv \right) \qquad ..2.4$$

5. Equation of state

$$P = \rho RT \qquad ..2.5$$

CONSERVATION FORM:

The conservation form of inviscid compressible flow eqns.was given by Courant and Friedrichs (1948) but Lax (1954) was the first to use this form to achieve a conservative finite difference scheme.

When the usual differential equations are rearranged so that the conservation variables ρ, ρu, ρv, e (to be defined) are the independent variables, then the use of conservative

finite difference methods assures identical conservation of mass, momentum and energy. The Rankine Hugnoit relations (2) for a normal shock wave are based on gross conservation and are independent of details within the shock structure. The conservation equations satisfy the Rankine Hugnoit relations and therefore produce correct jump across the shock. Longley (1960) tested four separate differencing methods and found that all produce correct shock speeds because conservation equations were used. Many subsequent calculations showed that conservation form is more accurate for calculations across the shock wave.

The conservation form can be obtained by manipulating the equations (2.1...2.5). The final form of the equation is as follows:

$$\frac{\partial U}{\partial t} + \frac{\partial F}{\partial x} + \frac{\partial G}{\partial y} = 0 \qquad ..2.6$$

Where U, F, G are defined as

$$U = \begin{vmatrix} \rho \\ \rho u \\ \rho v \\ e \end{vmatrix}$$

$$F = \begin{vmatrix} \rho u \\ \rho u^2 + p \\ \rho uv \\ \rho u h_o \end{vmatrix}$$

$$G = \begin{vmatrix} \rho v \\ \rho uv \\ \rho v^2 + p \\ \rho v h_o \end{vmatrix}$$

Thus we have Euler's equation (2.6) in conservation form.

Ref. (9)

Assuming that enthalpy remains constant i.e.

$$h_o = \left(\frac{\gamma}{\gamma - 1} \right) \frac{P}{\rho} + \frac{1}{2} \left(u^2 + v^2 \right) \qquad ..2.7$$

Thus the energy equation is eliminated from the solution and pressure can be directly determined from the above eqn.

Now by knowing the initial and final conditions for the flow eqn. (2.6) can be integrated to provide the inviscid solution at a later time level. Since steady state is a special

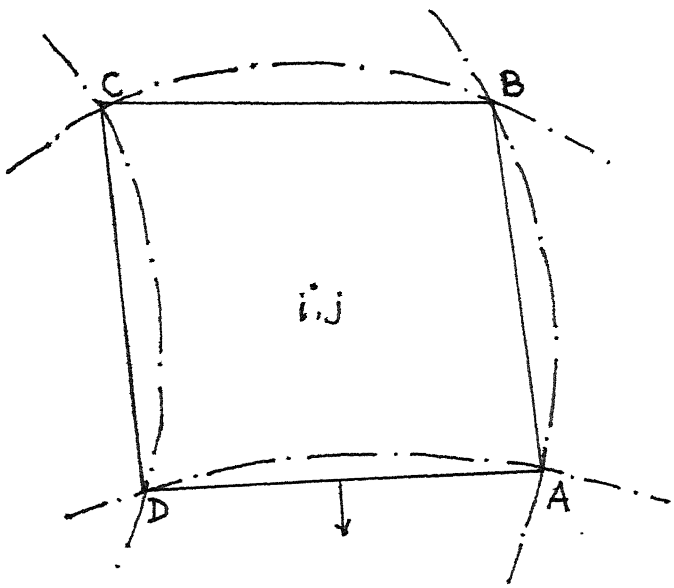

Fig- (2.1)

case of unsteady flow, the steady flow solution can also be obtained by solving the unsteady Euler eqn.

The eqns. may be solved in either Finite difference form or Finite volume form e.g. (Veuillot (1976), Gopalakrishna and Bozzala (1971)). It is usual to transform the computational domain into uniform rectangular grid and to express derivatives of flow variables in terms of values at the nodes of the grid. Specialized numerical techniques (Maccormack or Lax-Wendroff schemes are needed to ensure the stability of the integration of the equation through time until steady state is reached.

In finite volume method (McDonald (1971), Denton (1975)) the equations are regarded as equations for conservation of mass, energy and momentum applied to a set of interlocking control volumes formed by a grid in the physical plane. Thus in this way the conservation of all the three is ensured. This is not true in the differential approach. In the present analysis the Finite volume formulation is used.

FINITE VOLUME FORMULATION

The Finite volume cell method is based upon an integral form of equation to be solved. The computational region is divided into elementary quadrilateral volumes within which integration is carried out. Such a procedure allows one to deal with complicated geometry without considering the equation written in curvilinear coordinates. (This also preserves the property of conservation). Only the coordinates of the

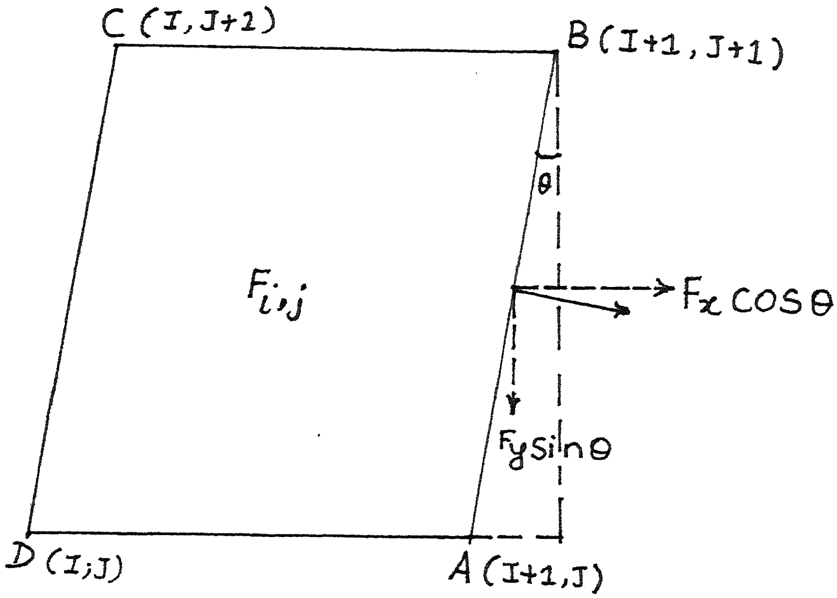

Fig: 2.2

ELEMENTRY CELL

corners of the cell are necessary. The technique is described by a model equation.

$$\frac{\partial U}{\partial t} + \frac{\partial F}{\partial x} + \frac{\partial G}{\partial y} = 0$$

The integral form of the above equation is

$$\int_A \frac{\partial U}{\partial t} dA + \int \nabla . H \, dA = 0$$

Assuming $\frac{\partial U}{\partial t}$ constant within the cell,

$$\frac{du}{dt} \int_A dA + \int_S H.ndS = 0 \qquad ..2.7$$

$$A\frac{du}{dt} + \int_S (F.1_x + G.1_y)dS = 0$$

Where A = Area of the cell

S=surface integral

n= unit normal to the boundary

If cartesian coordinates are used in the evaluation of boundary integral then we have,

$$H.ndS = Fdy - Gdx$$

Assuming a curvilinear mesh Fig (2.1), the elementary cell is quadrilateral ABCD. The eqn. 2.7 reduces to

$$\frac{dU}{dt} = (H_{AB} + H_{BC} + H_{CD} + H_{DA})/A \qquad ..2.8$$

Thus the eqn. 2.8 is an ordinary differential eqn., which describes the evolution of time average value of U.

The Fluxes H_{AB} , H_{BC} , H_{CD} , H_{DA} are defined as

$$H_{AB} = F_{AB} \Delta Y_{AB} - G_{AB} \Delta X_{AB}$$

$$H_{BC} = F_{BC} \Delta Y_{BC} - G_{BC} \Delta X_{BC}$$

$$H_{CD} = F_{CD} \Delta Y_{CD} - G_{CD} \Delta X_{CD}$$

$$H_{DA} = F_{DA} \Delta Y_{DA} - G_{DA} \Delta X_{DA}$$

where F_{AB}, F_{BC} , G_{AB} , G_{BC} are respectively values of F,G on sides AB , BC , CD , DA and where

$$\Delta X_{AB} = X_B - X_A$$

$$\Delta Y_{AB} = Y_B - Y_A$$

and so on respectively.

Let $F_{i,j}$ be the mean value of F in elementary cell, see Fig (2.2). The straight forward way to define the fluxes is

$$H_{AB} = \frac{1}{2} (F_{i+1,j} + F_{i,j}) Y_{AB} - \frac{1}{2} (G_{i+1,j} + G_{i,j}) X_{AB}$$

With analogous expressions for other fluxes. If uniform cartesian mesh is considered, these formulas yield the usual central differencing. In the same manner, it is possible to define a non-centered scheme by assuming, for instance, that the mean of F on AB is defined as weighted average.

$$\omega * F_{i+1,j} + (1-\omega) * F_{i,j} \quad \text{instead of} \quad \frac{1}{2} \left(F_{i+1,j} + F_{i,j} \right)$$

If u is positive then $\omega = 1$

If u is negative then $\omega = 0$

for average value $\omega = 0.5$

Such a definition with $\omega = 1$ or $\omega = 0.5$ is used in Finite volume methods suggested by Mac-cormack and Paullay (1972).

Ultimately eqn (2.8) reduces to

$$\frac{dU}{dt} = \left(H_{AB} + H_{BC} + H_{CD} + H_{DA} \right) / A$$

So to get the flow field the above eqn is to be solved by numerical methods. In the present analysis we are using Runge Kutta method.

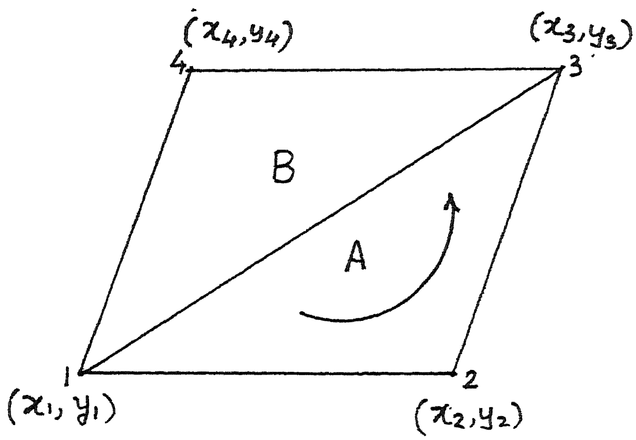

Fig: 3.1.

Chapter 3

Method of solution

The eqn to be solved is

$$\frac{dU}{dt} = (H_{AB} + H_{BC} + H_{CD} + H_{DA}) /A$$

For a particular geometry,a grid is generated over its whole domain. The values of all the variables are found at the center of each cell. Apply the unsteady method of choice to integrate the eqns forward in time subject to the inviscid flow boundary conditions of no flow normal to the surface and prescription of appropriate free stream conditions. In the present analysis we are using explicit shock capturing method.

To solve the eqns we require the values of fluxes and the Areas of each cell, which are computed as follows.

Method to find the area of the cell

It is derived from the area under the curve method. Consider a cell as shown in fig(3.1).Divide the cell into two triangles. Let (x_1,y_1) ,(x_2,y_2) be the coordinates of the vertices of the triangle,then the area of the triangle (1,2,3) given by

$$A=0.5*(x_1(y_2- y_3)+x_2(y_3- y_1)+x(y_1- y_2)) \qquad ..(3.1)$$

$$B=0.5*(x_1(y_3- y_4)+x_2(y_4- y_1)+x_3(y_1- y_3)) \qquad ..(3.2)$$

Thus the area of cell is given by the sum of A + B .Thus by this the area of all the cells can be found.

Now the second step is to find the fluxes.

$$U = \begin{vmatrix} \rho \\ \rho u \\ \rho v \\ e \end{vmatrix} \qquad F = \begin{vmatrix} \rho u \\ \rho u^2 + p \\ \rho uv \\ \rho uh \end{vmatrix}$$

and

$$G = \begin{vmatrix} \rho v \\ \rho uv \\ \rho v^2 + p \\ \rho v h_o \end{vmatrix}$$

Now let us represent them in terms of (I,J,K),where (I,J) are coordinate points and K=parameter.

$U(I,J,1) = \rho$

$U (I,J,2) = \rho u$

$U (I,J,3) = \rho uv$

Similarly the values of F and G can be written. The fourth parameter we are not considering as we have assumed constant enthalpy. So energy eqn does not come into account. Now to find u

$$u = U (I,J,2)/ U(I,J,1)$$

Similarly $v = U (I,J,3)/ U(I,J,1)$

So once ρ, u, and v are determined then p can be found by the eqn

$$p = \frac{\rho}{\gamma} \left[\gamma R T_o - \frac{(\gamma - 1)}{2}(u^2 + v^2)\right]$$

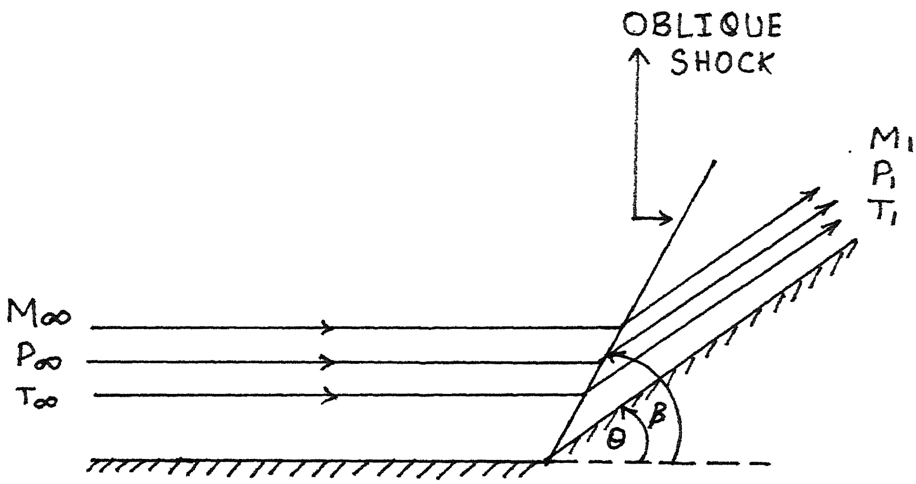

Fig:(3.2) FLOW OVER A WEDGE.

Thus by knowing the values of u ,v, ρ ,and p the fluxes H_{AB} can be calculated as defined earlier. Thus now applying proper boundary conditions the problem is solved.

3.1 Test case (Flow over a wedge)

Fig: 3.2

The free stream conditions are

1. Mach number = 2
2. Pressure = 1 bar
3. γ = 1.4
4. R =287 J/ kg. K

The unknowns to be found in the entire field

1. ρ - density
2. u - velocity components in x
3. v - and y directions.
4. p - pressure

When a supersonic flow passes over an compression corner a shock is formed. After the shock the Mach number decreases and the pressure, density and temperature rises discontinuously. The shock angle β can be found from θ-β-M relations, Ref (4). Presently we have used Maccormack relationsto find the density, velocity, pressure, and Mach number after the shock.

$P_o = 1$, $\rho_o = 1$, $M_o > 1$

Eqns are in non dimensional form.

$$\frac{P_1}{P_o} = \frac{2\gamma M_1 \sin \beta^2_{shk} - (\gamma - 1)}{\gamma + 1} \qquad ..(3.1.1)$$

$$\frac{\rho_1}{\rho_o} = \frac{(\gamma+1) Mo^2 \, Sin^2 \beta_{shk}}{(\gamma+1) Mo^2 \, Sin^2 \beta_{shk} + 2} \qquad ..(3.1.2)$$

$$\frac{M_1}{M_o} = \sqrt{\frac{(\gamma - 1) Mo \; Sin \; \beta_{shk} + 2}{Sin \; (\beta_{shk} - \theta^2) \; 2\gamma Mo \; Sin \beta_{shk} \, {}^2 (\gamma - 1)}} \qquad ..(3.1.3)$$

Computational mesh

$Dx = \Delta x = 1/16$

$Dy = \Delta y = 2 * Dx * (\tan\beta + u\phi + c\phi)/(u\phi + c\phi)$

$c\phi = \sqrt{\frac{\gamma \, . P}{\rho}}$

$u\phi = M\infty * c\phi$

Thus the step length Δx and Δy are found.

Now the time step is given by

$$Dt = \text{Smaller value of } \frac{(Dx \; , \; Dy)}{u\phi + c\phi}$$

This is due to Courant, Friedrichs, and Levy (1928) stability restriction.

BOUNDARY CONDITIONS

The boundary conditions involved in solving this problem are as

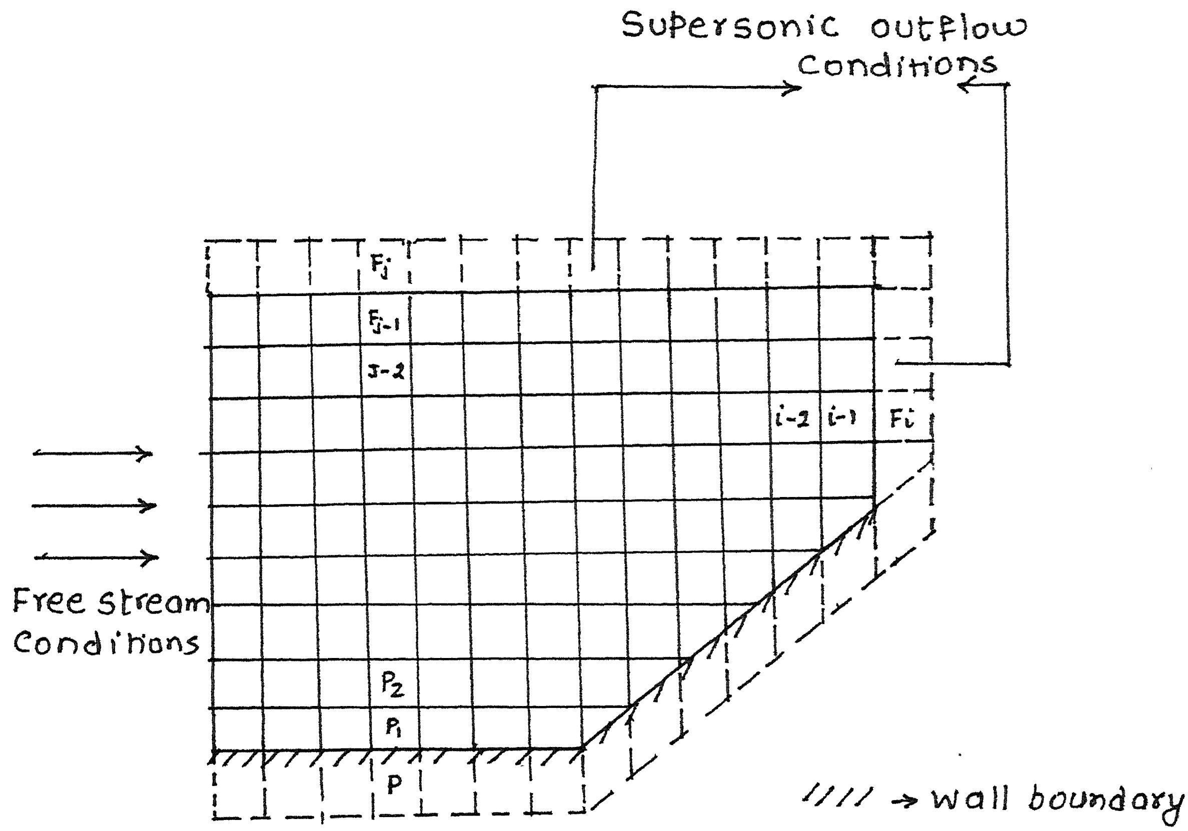
Supersonic outflow conditions
F_j
F_{j-1}
j-2
i-2
i-1
Fi
Free stream conditions
P_2
P_1
P
//// → wall boundary

Fig: 3-3.

follows:

Inlet, exit, and solid wall boundary conditions are enforced during calculation of fluxes through the faces situated on the edge of the computation domain. For solid walls the mass flux through the wall vanishes (q.n =0).The only contribution to the wall flux is the pressure acting on the wall,which is obtained by the second order extrapolation using the averages and the gradients of the cells adjacent to the walls.

Fluxes through the inlet and the exit planes are obtained using ghost points outside the computational domain. For supersonic intakes the ghost point values are set at inlet conditions,where-as for supersonic exit all quantities are extrapolated from the interior. For subsonic inflow,entropy, total enthalpy and flow angles are specified, while pressure is extrapolated to the inlet plane.(Using averages and the gradients of the first cell inside the domain).For subsonic outflow static pressure is specified and entropy, total, enthalpy and flow angle are extrapolated. Ref. (10).

Mathematical representation of B.C

There are four boundary conditions Ref Fig (3.3)

1. Free stream boundary condition
2. Wall boundary condition
3. Two supersonic out flow conditions

1. Free stream B.C.

For supersonic case fix all the variables i.e,Fix ρ, u, v, p .
For subsonic case fix Po ,To and every time calculate ρ, u, v, p . Flow at the inlet is of free stream case. Thus we know all the variables at the inlet and we can find the changes in flow as time step changes.

2 . Wall B.C

There is no flux in the direction normal to the wall. This is proved by an example, If we have a curved surface velocity in normal dirdirection is zero, i.e. V.n = 0 Let l_x and l_y be the direction cosines then $ul_x + vl_y = 0$. In our case the total flux is $\int$ F.nds where F= (F,G) and F,G are defined earlier. So,

$$
\begin{aligned}
Fl_x + Gl_y &= \rho u l_x + \rho v l_y \\
&= \rho(ul_x + vl_y) \\
&= 0 \qquad \text{as } (ul_x + vl_y = 0) \\
&= (\rho u^2 + p)l_x + (\rho uv) \\
&= \rho u(ul_x + vl_y) + pl_x \\
&= pl_x
\end{aligned}
$$

Similarly $(\rho uv)l_x + (\rho v^2 + p)l_y$

$$= pl_y$$

Thus we get

$$F(I,J,1) = 0 \qquad G(I,J,1) = 0$$

$$F(I,J,2) = p \qquad G(I,J,2) = 0$$

$$F(I,J,3) = 0 \qquad G(I,J,3) = 0$$

Thus at wall only pressure term is there and rest all the fluxes are zero. The pressure is found by linear extrapolation.

Outflow B.C (at exit)

In case of supersonic flow it is just the linear extrapolation.

From the central difference formula $F_i = \dfrac{F_{i-1} + F_{i+1}}{2}$

$$F_{i+1} = 2^* F_i - F_{i-1}$$

$$F_i = 2^* F_{i-1} - F_{i-2}$$

Thus the value is extrapolated by using the values of two previous cells.

outflow B.C (at top)

It is assumed that the outflow is supersonic and linear extrapolation is used.

$$F_j = \frac{F_{j-1} + F_{j+1}}{2}$$

$$F_j = 2^* F_{j-1} - F_{j-2}$$

Thus the fluxes at all points including the boundaries can be found.

RUNGE KUTTA METHOD

After finding all the fluxes we substitute in the differential eqn. It being a simple first order differential eqn. Runge kutta method is used to solve it. We are using fourth order Runge kutta method for solving the differential eqn (3.1). It gives higher time step by $2\sqrt{2}$ times theoretically and also the error in computation reduces. The error graph falls sharply as the order increases, but above fourth order the changes are less compared to the computing time.

The routine is written as:

$$U^0 = U^n$$

$$U^1 = U^0 - 1/4 * \Delta t\ R^0$$

$$U^2 = U^0 - 1/3 * \Delta t\ R^1$$

$$U^3 = U^0 - 1/2 * \Delta t\ R^2$$

$$U^4 = U^0 - \Delta t\ R^3$$

$$U^{n+1} = U^4$$

Here U^n and U^{n+1} are values at the beginning and the end of n^{th} time step.

CHAPTER 4

DISSIPATION

As we go for higher Mach numbers of the order of Mach 5 it is found that there are many oscillations and solution instead of converging just blows up. In such applications it is often necessary to add an artificial viscosity (dissipation) term to the schemes described before. Such a term is necessary to:

1. Select solution satisfying the entropy condition.
2. Prevent nonlinear instabilities.
3. To damp spurious oscillations of numerical solution.

The concept of introducing an artificial term to smear out shocks was first introduced by von Neumann (1944).It was so devised that it was effective only in the regions where high compressions occur,i.e where shocks are building up. Away from the shock region ,the effect is negligible. The Euler eqns. are nondissipative hence it is desirable to have a scheme with minimum dissipation,for higher accuracy.

In the present case there were many oscillations so some dissipation was added,to suppress the odd and even point oscillation. The damping added is proportional to difference between local value of enthalpy and the free stream value. Thus the Euler eqn becomes:

$$A_{ij}\frac{dU}{dt} + Q_{ijk} - D_{ijk} = 0 \qquad \text{or}$$

$$\frac{dU}{dt} = -\frac{1}{A}\left(Q_{ijk} - D_{ijk} \right) /A$$

In x direction

$$V_{i,j} = K^{(2)} \frac{\left| P_{i+1,j} - 2P_{i,j} + P_{i-1,j} \right|}{\left| P_{i+1,j} \right| + 2\left| P_{i,j} \right| + \left| P_{i-1,j} \right|}$$

$$V_{i+1,j} = K^{(2)} \frac{\left| P_{i+2,j} - 2P_{i+1,j} + P_{i,j} \right|}{\left| P_{i+2,j} \right| + 2\left| P_{i+1,j} \right| + \left| P_{i,j} \right|}$$

$$V_{i+2,j} = K^{(2)} \frac{\left| P_{i+3,j} - 2P_{i+2,j} + P_{i+1,j} \right|}{\left| P_{i+3,j} \right| + 2\left| P_{i+2,j} \right| + \left| P_{i+1,j} \right|}$$

$$V_{i-1,j} = K^{(2)} \frac{\left| P_{i,j} - 2P_{i-1,j} + P_{i-2,j} \right|}{\left| P_{i,j} \right| + 2\left| P_{i-1,j} \right| + \left| P_{i-2,j} \right|}$$

$$V_{i-2,j} = K^{(2)} \frac{\left| P_{i-1,j} - 2P_{i-2,j} + P_{i-3,j} \right|}{\left| P_{i-1,j} \right| + \left| 2P_{i-2,j} \right| + \left| P_{i-3,j} \right|}$$

In y direction:

$$V_{i,j} = \frac{\left| P_{i,j+1} - 2P_{i,j} + P_{i,j-1} \right|}{\left| P_{i,j+1} \right| + 2\left| P_{i,j} \right| + \left| P_{i,j-1} \right|}$$

$$V_{i,j+2} = \frac{\left| P_{i,j+2} - 2P_{i,j+1} + P_{i,j} \right|}{\left| P_{i,j+2} \right| + 2\left| P_{i,j+1} \right| - \left| P_{i,j} \right|}$$

$$V_{i,j+3} = \frac{\left| P_{i,j+3} - 2P_{i,j+2} + P_{i,j+1} \right|}{\left| P_{i,j+3} \right| + 2\left| P_{i,j+2} \right| + \left| P_{i,j+1} \right|}$$

$$V_{i,j-1} = \frac{\left| P_{i,j} - 2P_{i,j-1} + P_{i,j-2} \right|}{\left| P_{i,j} \right| + 2\left| P_{i,j-1} \right| + \left| P_{i,j-2} \right|}$$

$$V_{i,j-2} = \frac{\left| P_{i,j-1} - 2P_{i,j-2} + P_{i,j-3} \right|}{\left| P_{i,j-1} \right| + 2\left| P_{i,j-2} \right| + \left| P_{i,j-3} \right|}$$

$$E^{(2)}_{i+1/2,j} = 1.0*AMAX(V_{i+1,j}, V_{i,j})$$

$$E^{(2)}_{i-1/2,j} = 1.0*AMAX(V_{i,j}, V_{i-2,j})$$

$$E^{(2)}_{i,j+1/2} = 1.0*AMAX(V_{i,j+1}, V_{i,j})$$

$$E^{(2)}_{i,j-1/2} = 1.0*AMAX(V_{i,j}, V_{i\ j-1})$$

$$E^{(4)}_{i+1/2,j} = MAX(0, (1/32 -E_{i+1/2,j}))$$

$$E^{(4)}_{i-1/2,j} = MAX(0, (1/3 - E^{(2)}_{i-1/2,j}))$$

$$E^{(4)}_{i,j+1/2} = MAX(0,(1/32 - E^{(2)}_{i,j+1/2}))$$

$$E^{(4)}_{i,j-1/2} = MAX(0,(1/32 - E^{(2)}_{i,j-1/2}))$$

The dissipation operator which we used is a combination of second and fourth order differences. These are done separately in X and Y directions.

Ex for density eqn

$$D\rho = D_x\rho + D_y$$

$$D_x\rho = d_{i+1/2,j} - d_{i-1/2,j}$$

$$D_y\rho = d_{i,j+1/2} - d_{i,j-1/2}$$

We have

$$d_{i+1/2,j} = A_{i+1/2,j} \left[E^{(2)}_{i+1/2,j} (\rho_{i+1,j} - \rho_{i,j}) \right.$$
$$\left. - E^{(4)}_{i\ 1/2\ j} (\rho_{i+2,j} - 3\rho_{i+1,j} + 3\rho_{i,j} - \rho_{i-1,j}) \right]$$

$$d_{i-1/2,j} = \frac{A_{i-1/2,j}}{t} \left[E^{(2)}_{i-1/2,j} (\rho_{i,j} - \rho_{i-1,j}) \right.$$
$$\left. - E^{(4)}_{i-1/2,j} (\rho_{i+1,j} - 3\rho_{i,j} + 3\rho_{i-1,j} - \rho_{i-2,j}) \right]$$

$$d_{i,j+1/2} = \frac{A_{i,j+1/2}}{t} \left[E^{(2)}_{i,j+1/2} (\rho_{i,j+1} - \rho_{i,j}) \right.$$
$$\left. - E^{(4)}_{i,j+1/2} (\rho_{i,j+2} - 3\rho_{i,j+1} + 3\rho_{i,j} - \rho_{i,j-1}) \right]$$

$$d_{i,j-1/2} = \frac{A_{i\ j-1/2}}{t} \left[E^{(2)}_{i,j-1/2} (\rho_{i,j} - \rho_{i,j-1}) \right.$$
$$\left. - E^{(2)}_{i,j-1/2} (\rho_{i,j+1} - 3\rho_{i,j} + 3\rho_{i,j-1} - \rho_{i,j-2}) \right]$$

$$E^{(2)}_{i+1/2,j} = \mathrm{MAX}(V_{i+1,j} , V_{i,j})$$

In the above eqns $K^{(2)}$ is a constant, however the outcome is

positive ,of the order (x^2), and proportional to the second difference of pressure. It does seem to do a nice job suppressing the oscillation around shocks.

$$E^{(4)}_{i\,1/2\,j} = \mathrm{MAX}(\,0,(\,K^{(4)}, - E^{(2)}_{i+/2\,,j}\,))$$

where $K^{(4)}$ is second constant.

CHAPTER 5

RESULTS AND DISCUSSION

WEDGE FLOW CASE

This problem was taken as a trial case to test whether the algorithm is working well or not. This case is taken from Ref.(4) figure as shown in (3.2).The initial condition is uniform flow. The wedge dimensions and the free stream conditions are described earlier. The wedge angle is 4.5^{0}. firstly a computational mesh is generated over the entire wedge and values of each variable are found at the node points,this goes on till the steady state is reached.

The analysis was carried by two different grids, one, mesh along the shock wave and other perpendicular to the wedge, for Mach 2 flow. According to the standard test data available when mesh is aligned to the shock wave the density or the pressure variation along the wedge is as shown in fig (5.6 - 5.10).till the shock the density is constant and sudden jump at the location of the shock and again constant over the entire length of the wedge. After computing the results,they were found satisfactory.

When the grid was made perpendicular there were many oscillations in the shock region and, then uniform as the shock strength reduces. Usually this type of grid is used in solving the practical problems,whereas the former is the ideal case and

useful for a particular problem where shock location and shock angle can easily be found out, i.e for simple geometries.

For the wedge flow case the computation was done for two different cases: one ω = Ø.5 i.e.(central differencing) and other ω = 1 i.e. upwinding. In the former case there were wide oscillations (odd and even point)and the curve has saw tooth type profile. Fig(5.9). With upwinding the oscillations were damped to certain extent and results were slightly nearer to the ideal case. With upwinding the resulting finite difference eqn is only of first order accuracy,thus accuracy reduces slightly.

To damp the oscillations further an artificial viscosity term (dissipation) was added along with upwinding and we got a smooth curve (ideal curve).So due to upwinding and dissipation the oscillations were fully damped .The results are shown in graphs (5.6 - 5.1Ø).The curve trends were found to be same when tried for different Mach numbers.

The intake geometry is as shown in the fig (1.1) and the size and the initial conditions are already specified in chapter 1. The physical domain was transformed into computational domain. A grid size of 55*1Ø was used in calculations. He grid was spaced equally and in same concentration everywhere.

To start with free stream conditions were assumed initially at all grid points except near boundaries where

appropriate boundary conditions were used. Only the top boundary condition was changed i.e. wall boundary condition was introduced instead of supersonic outflow boundary condition compared to wedge case as discussed earlier, rest all boundary conditions remain same.

The time step was determined by C.F.L restriction and integrated with time still steady state is reached. To access the quality of the solution the pressure profiles were compared at three locations on upper and lower walls. These lactations lie ahead,aft of,and in the separation region. Fig(5.4) and (5.5) show the pressuredistribution on upper and lower walls. When compared with Ajaykumar's Ref.(4.) solution the trend was found to be same but the numerical values were slightly differing. This may be mainly due to the efficient implicit and explicit (combined)method used by him ,this method gives higher time step. secondly he used a 51 * 51 grid which gives higher resolution. Lastly he was dealing withaviscous problem so the amount of damping used may be different. He must have also used the grid refinement i.e close grids near the shocks.

The pressure distributions at different stations were compared and the curve trends were found to be same.

CONCLUSIONS

With the above results we can conclude that that the results obtained by inviscid flow solution ...

the inviscid flow solution first in order to establish the correctness of the algorithm which will form an integral part of general case as well as to have the preliminary results with much less effort in comparison with the general case. The extension of this analysis for the general realflow situation could not be taken up due to limited availability of time for thesis work.

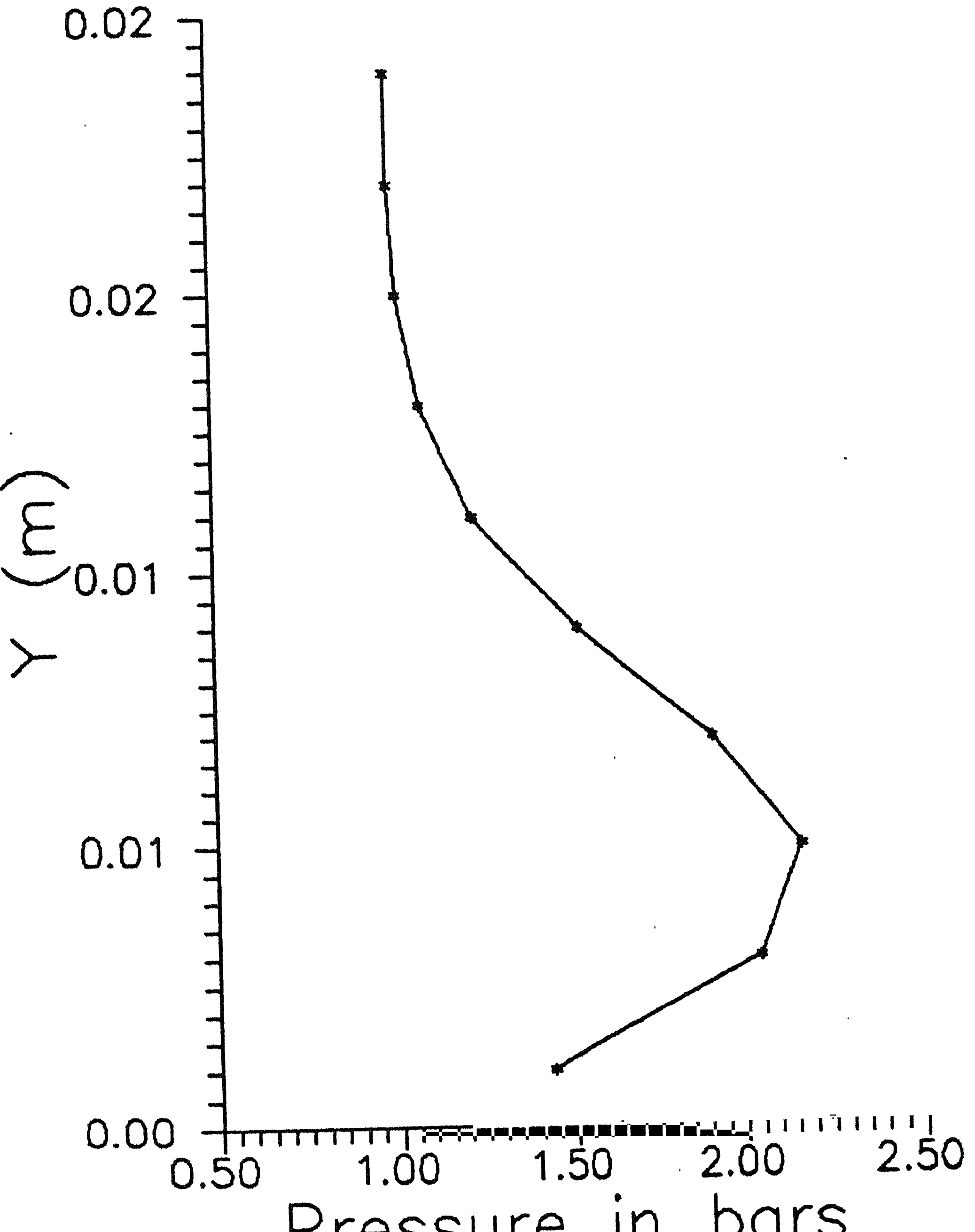

Fig. 5.1 – Pressure profile at x=0.05m

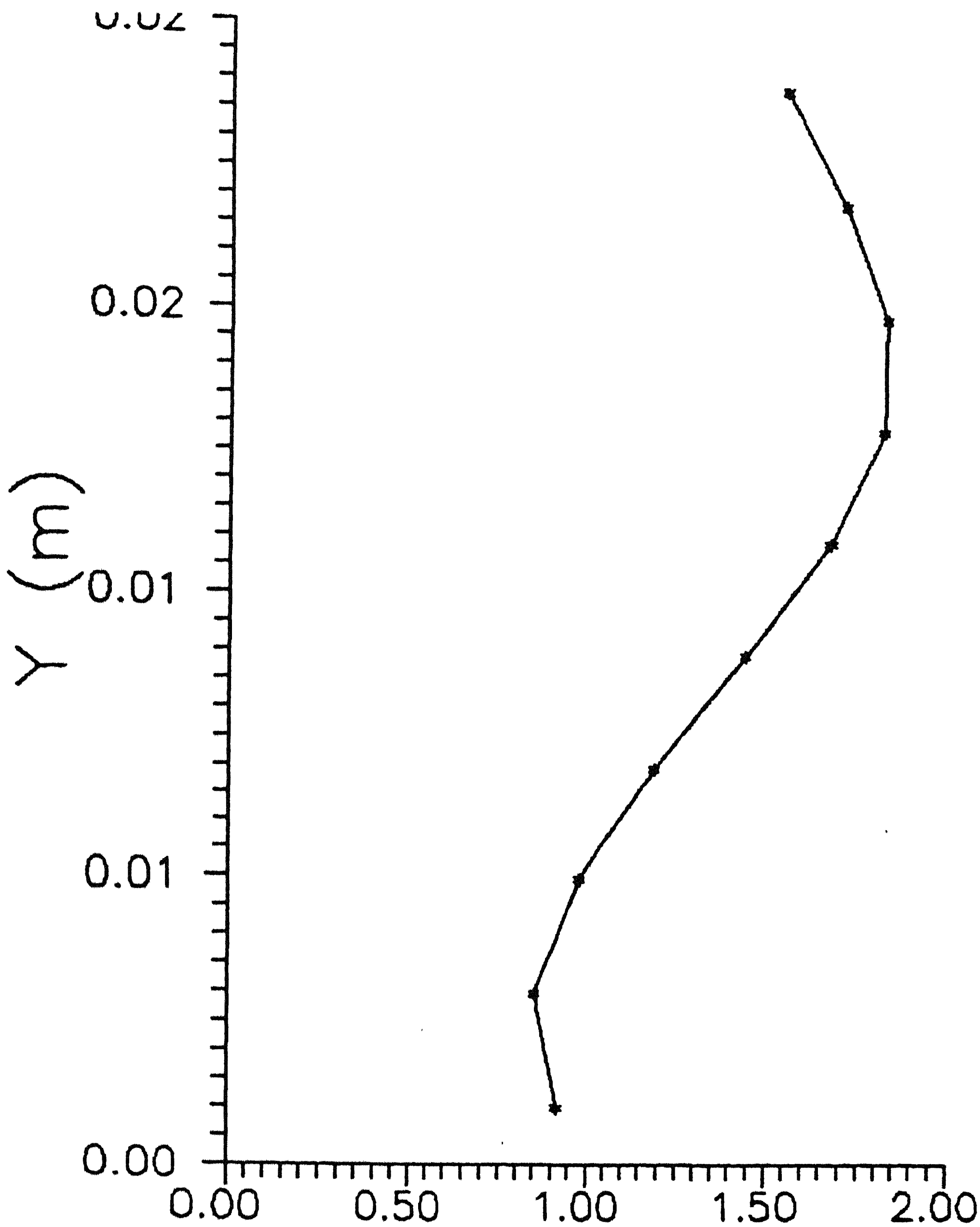

Fig. 5.2 —Pressure profile at x=0.074m

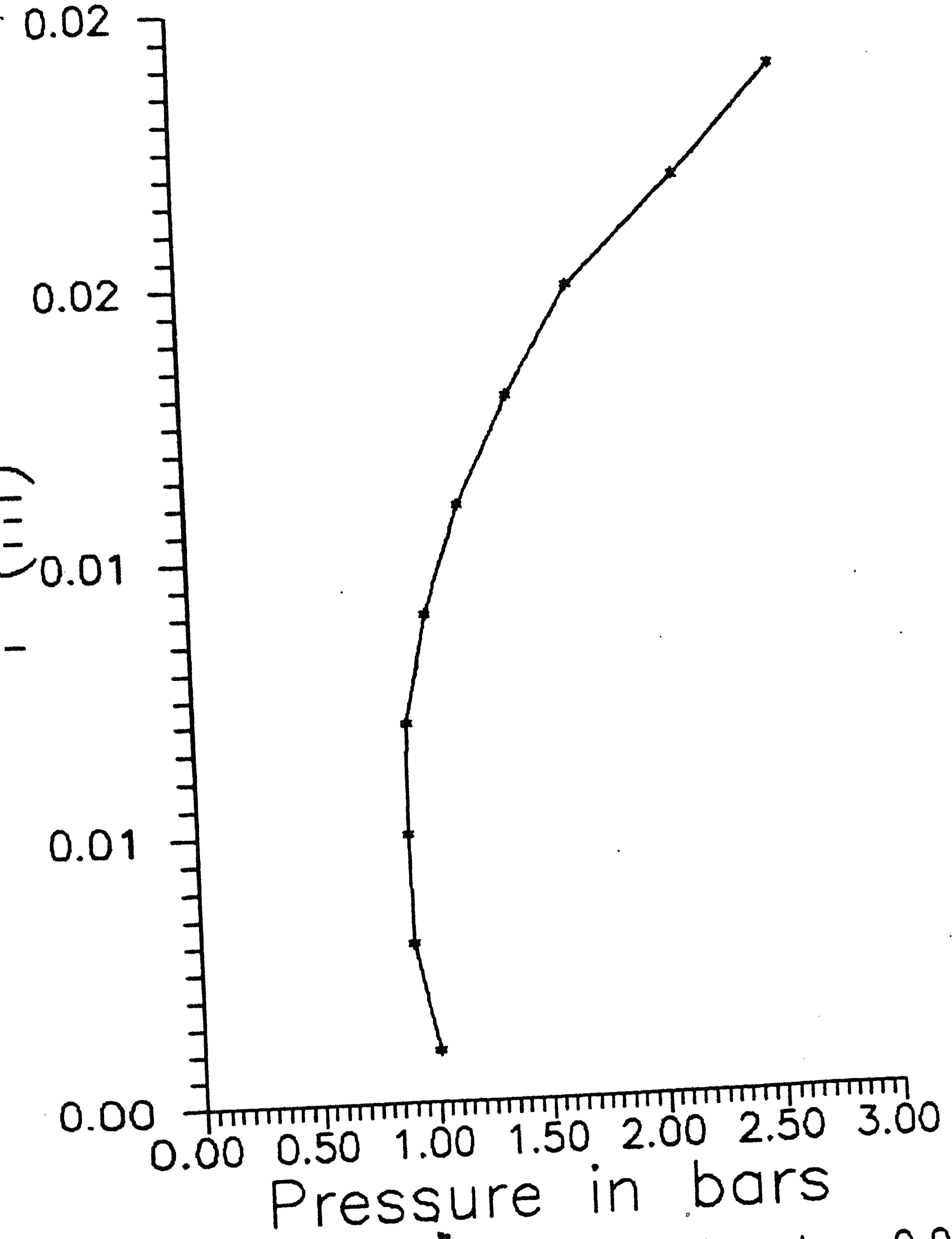

Fig. 5.3 –Pressure profile at x=0.088m

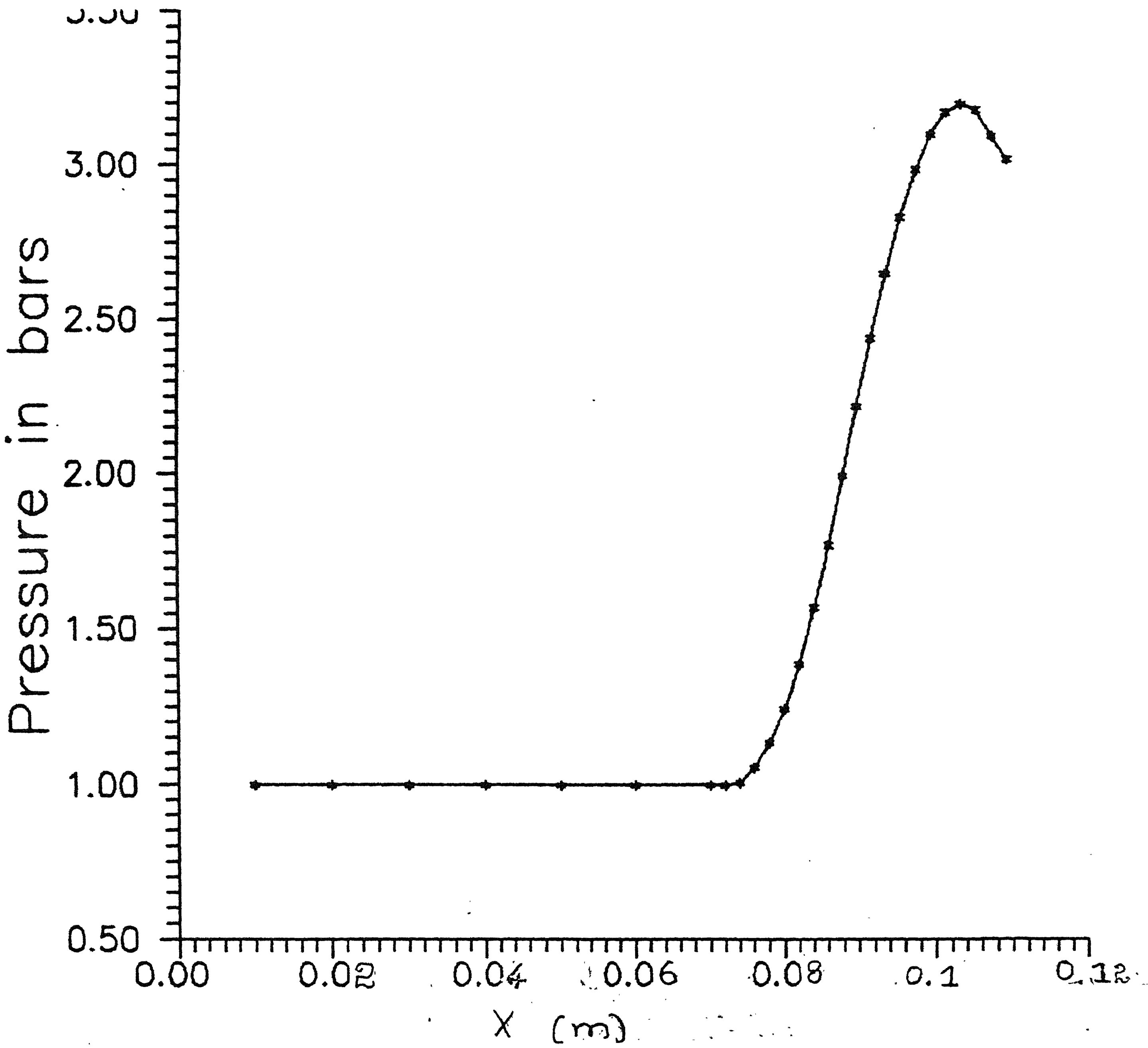

Fig. 5.4 —Upper wall pressure distribution

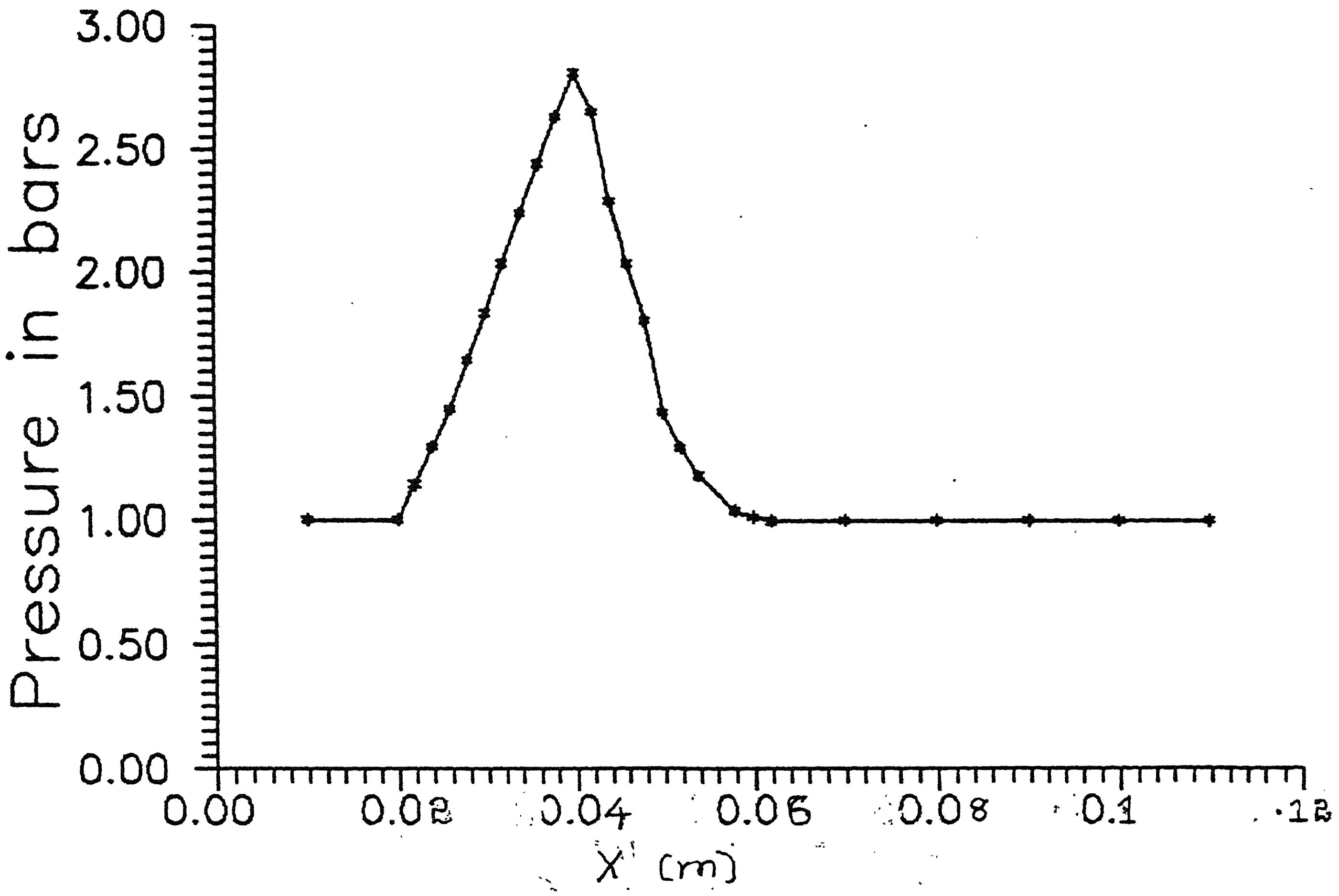

Fig. 5.5 -LOwer wall pressure distribution

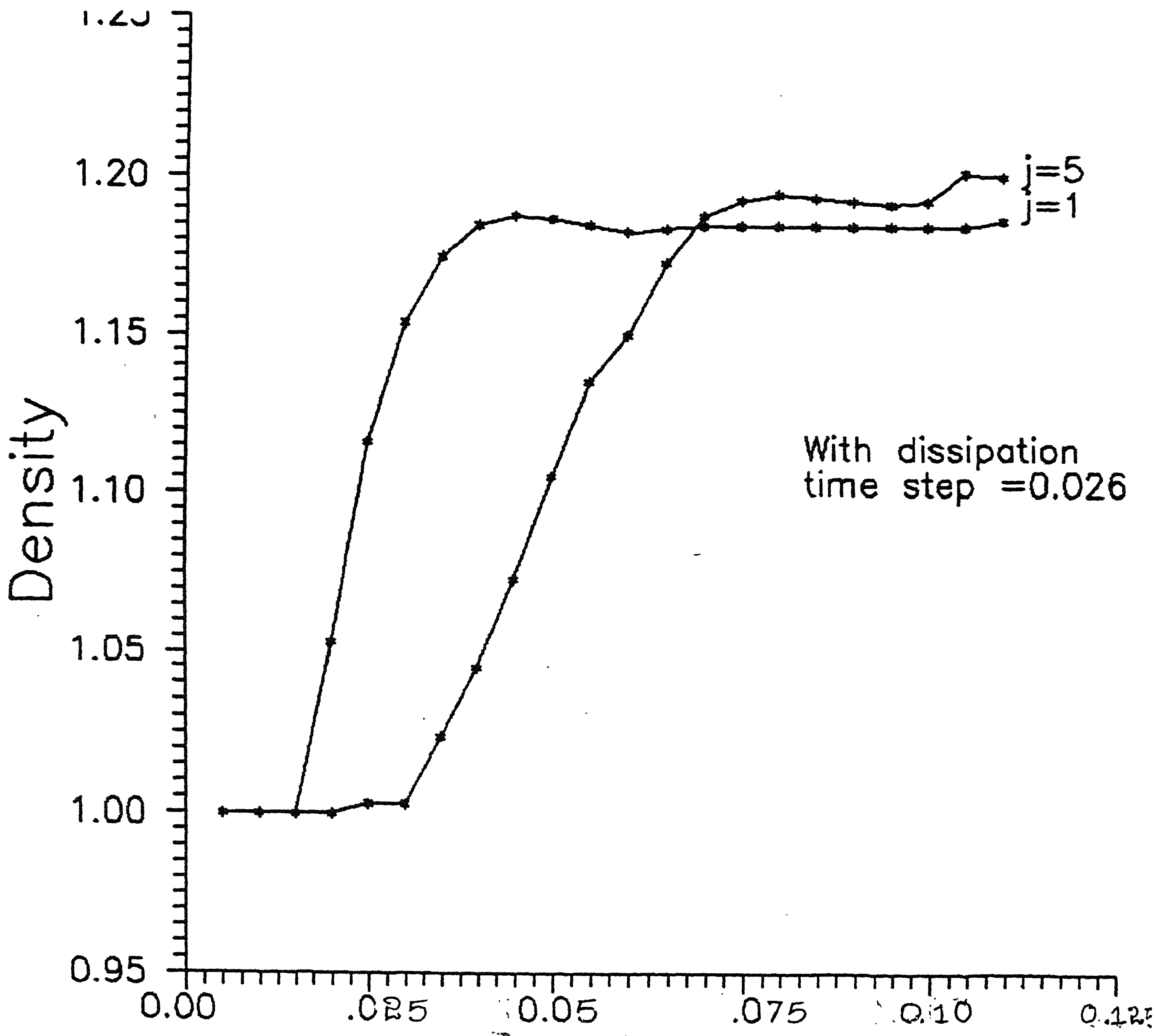

Fig 5.6 Density variation along the Length of wedge.

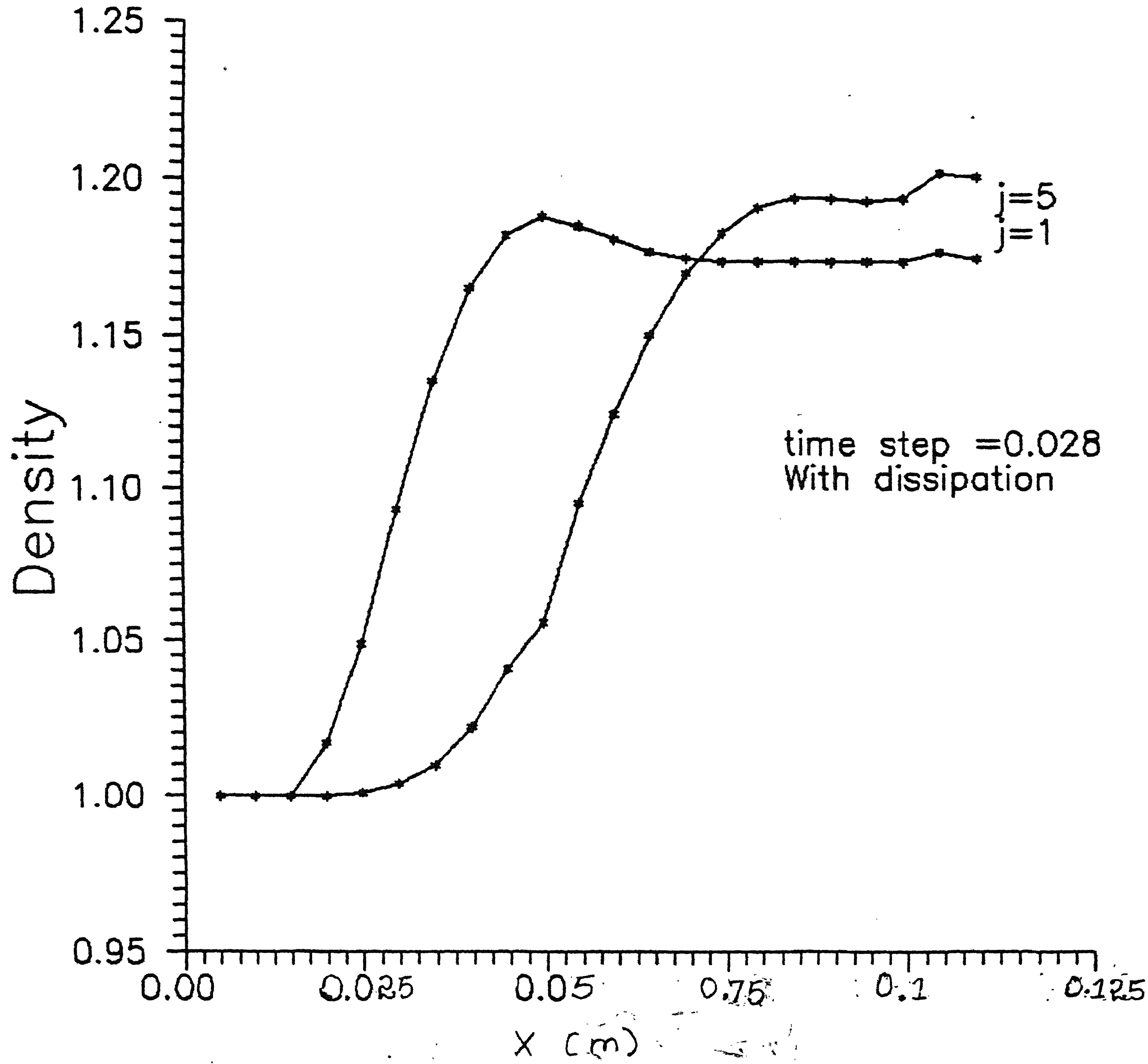

Fig. 5.7 – (Aligned grid)
Density variation along the length of wedge.

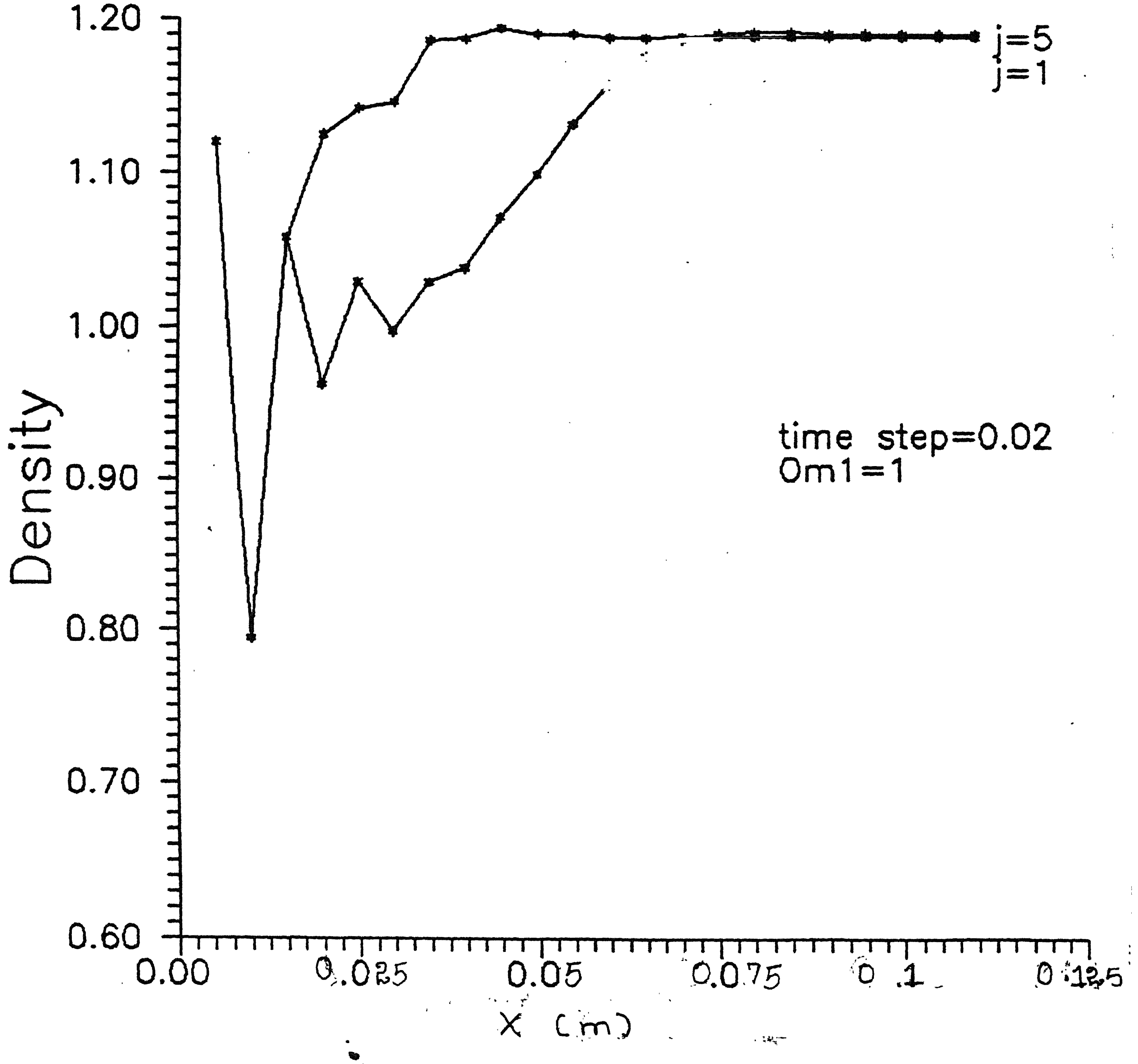

Fig. 5.8 (Without dissipation)
Density variation along the length of wedge

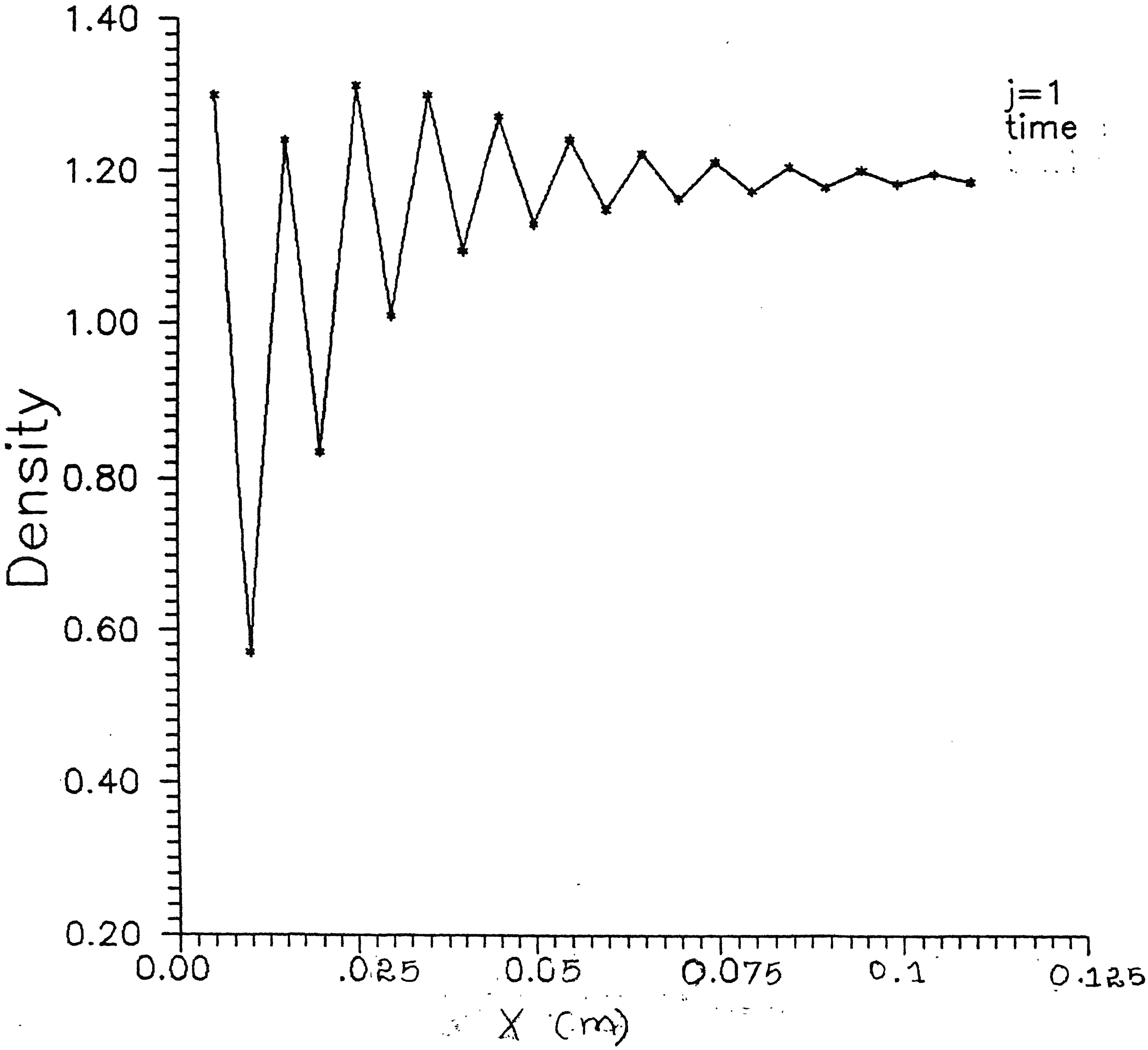

Fig. 5.9 – Density Variation along the Length of wedge.

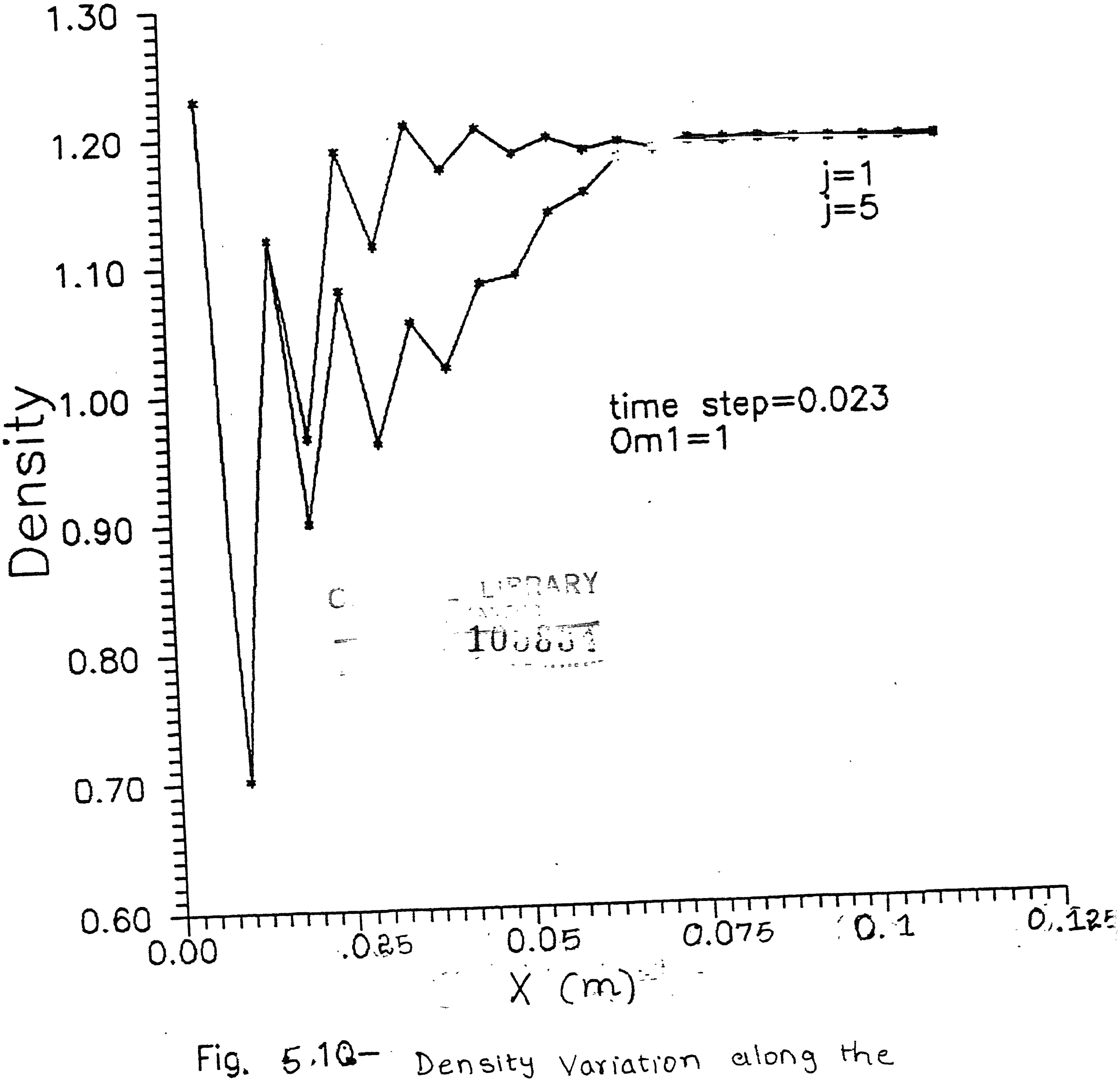

Fig. 5.10– Density Variation along the length of wedge

BIBILOGRAPHY

1. Anderson, J.D., "Fundamentals Of Aerodynamics",Int. to nunerical techniques for nonlinear supersonic flow. pp 450-478

2. Peyret,R.,and Taylor, D. "Computational Methods in Fluid Flow", Springer series in Computational physics. pp. 108-119

3. Ajay kumar,:Some observations on a New Numerical method for Solving Navier Stokes Equations:, NASA. TP, Nov. 1981 .

4. Maccormack, R.W., "Numerical Methods for ompressible Viscous flows", AIAA -81- 0110, JAN. 1981.

5. Jayachandran, T.and Mehta, R.C., "Numerical Solution Of Full Potential and Euler Equation", VSSC. Trivandrum.

6. Jameson, A.,Schmidt, E.,and Turkel, "Numerical Solution of Euler Equations by Finite Volume Methods using Runge Kutta Time Stepping Schemes", AIAA. 81 - 1259, JUNE 1981.

7. Ni, R.H., "A Multiple Grid Scheme for Solving Euler Equations", AIAA. Vol. 20, p. 1565-1571, Nov. 1981.

8. Roach, P.J., "Computational Fluid Dynamics", 1971.

9. Allmaras, S.R., and Giles, M.B., "A Second order Flux Split Scheme for Unsteady 2-D. Euler Equations on arbitrary meshes", AIAA. 1987

10. SIxth International Symposium on Air Breathing Engines, june 6-10. 1983, Paris,France.